5/A
C. Brown
8/14/03

TITLE OF INVENTION

Name: Gary John Corey

Citizenship: United States of America

Residence: 21445 Bundy Canyon Drive, Wildomar, California, USA, 92595

Title of Invention: Multi-Axes Tool Compensation -- 3D and 5-axis real-time interactive tool compensation inside the CNC machine tool controller.

CROSS-REFERENCE TO RELATED APPLICATIONS: Not Applicable

STATEMENT REGARDING FEDERALLY SPONSORED RESEARCH OR DEVELOPMENT: Not Applicable

REFERENCE TO A COMPUTER PROGRAM LISTING COMPACT DISK

APPENDIX

The Appendix contains two copies on compact disk of the entire Multi-Axes Tool Compensation computer program listing in standard ASCII character file format. Each compact disk contains the same single file entitled MTC.TXT.

BACKGROUND OF THE INVENTION

Multi-Axes Tool Compensation for 3D and 5-axis real-time interactive tool compensation relates to CNC machining techniques that are already in common practice for 2D tool compensation in CNC machining.

The applicable US patent Classification Definition to my invention is found in category 700 DATA PROCESSING: GENERIC CONTROL SYSTEMS OR SPECIFIC APPLICATIONS for Sub Class 1 GENERIC CONTROL SYSTEM, APPARATUS OR PROCESS.

Below are references to specific documents, articles and technical meetings related to my invention.

[1] STORI, J. A. and P. K. WRIGHT. A constant engagement offset for 2-1/2 D tool path generation. Proc. 1998 Intl. Mech. Engr. Congr. and Expo. (Anaheim, Calif., Nov. 1998) MED-vol. 8, 475-481 (1998).

[2] DeVOR, R. E., S. G. KAPOOR, R. ZHU, K. JACOBUS, I. LAZOGLU, S. SASTRY, and M. VOGLER. Development of mechanistic models for the prediction of machining performance: Applications to process and product quality. Proc. CIRP Intl. Wkshp. on Modeling of Machining Oper. (Atlanta, Ga., May 1998) 407-416 (1998).

[3] FLORES, M. A. and T-C. TSAO. Supervisory machining control implementation using an open architecture CNC. Proc. Japan-USA Symp. on Flexible Automat. (Otsu, Japan, Jul. 1998) 1157-1164 (1998).

[4] GAJJELA, R. R., S. G. KAPOOR, and R. E. DeVOR. A mechanistic force model for contour turning. 1998 Intl. Mech. Engr. Congr. and Expo. (Anaheim, Calif., Nov. 1998) 8, 149-159 (1998).

[5] KAPOOR, S. G., R. E. DeVOR, R. ZHU, R. GAJJELA, G. PARAKKAL, and D. SMITHEY. Development of mechanistic models for the prediction of machining performance: model-building methodology. Proc. CIRP Intl. Wkshp. on Modeling of Machining Oper. (Atlanta, Ga., May 1998) 109-120 (1998).

[6] KUMAR, P. and P. M. FERREIRA. Hierarchical control of flexibly automated manufacturing systems. Proc. Japan-USA Symp. on Flex. Automat. (Otsu, Japan, Jul. 1998) III, 1207-1214 (1998).

[7] K. KOTHARDARAMAN, KUMAR, P., and P. M. FERREIRA. Scalable, maximally-permissive deadlock avoidance for FMS. IEEE Conf. on Robotics and Automat. (Lèuver, Belgium, May 1998).

Other references:

Pro/MFG Technical Meeting at the Omni Rosen Hotel, Orlando, Florida
June 17, 2000 Compiled by Gene J Maes

Attendees:

Gene J Maes, Los Alamos National Laboratory,
Sam Moses, Caterpillar Inc.,
Marcus Vasquez, Solar Turbines,
Val Hubbard, Raytheon,
Jason Anderson, Adept Limited,
Bryan Garvin, Moen Inc.,
Brad Baas, Los Alamos National Laboratory,
Pete Lord, PTC,
Francois Lamy, PTC,

Charles Farah, PTC,
Gary Whalen, Harley-Davidson,
Tom Calenberg, MSC Technologies, Inc.,
Charles Wenning, Midmark Corp.,
Richard Bridy, ITT Goulds Pumps,
Johanna Rock, ITT Goulds Pumps,
Dan Schurr, Steelcase Inc.,
Brad Bush, kodak,
Chad Weber, John Deere,
Ron Johnson, United Defense,
Steve Wall, United Defense,
Randy Hoffman, Solar Turbines,
Norm Lamar, San Diego State University,
Jim Burns, San Diego State University,
Jeff Rowe, Honeywell,
Harold W. Kaiser, Walter Information Systems,
Anatol Borejdo, Walter Information Systems

Presentations by:

Charles Farah
Norm Lamar& Jim Burns (San Diego State University)
Glenn Coleman
Francois Lamy

BRIEF SUMMARY OF THE INVENTION

The object of Multi-Axes Tool Compensation is to provide CNC machinists using CNC controllers a convenient method for applying 3D and 5-axis tool compensation in real time directly within the CNC controller as they now enjoy when using the traditional 2D tool comp standards G41 and G42. Before my invention CNC controllers have not been technically advanced enough to employ multi-axes tool compensation methods. Using the technology methods of my invention for multi-axes tool compensation, the machine operator now has a pre-defined method to assign 3D and 5-axis tool characteristics at the CNC controller. CNC programmers and machinists now have the tools to issue 3D and 5-axis tool comp commands, which have not been available in traditional CNC controllers.

The CNC machine operator will no longer require the assistance of the CNC programmer to re-create a brand new CNC G code Program with new tool positions and definitions when a change is made. My invention allows the CNC machine operator to easily define the new tools using my complex 3D and 5-axis tool compensation algorithms built into the CNC Controller. These algorithms also provide for automatic tool gouge avoidance protection.

BRIEF DESCRIPTIONS OF THE SEVERAL VIEWS OF THE DRAWINGS

FIG 1. is the operator interface screen that users use to enter in the tool size and offsets.

FIG 2. individually labels the various dimensions used in the calculations of Multiple-axis Tool Compensation.

FIG 3. isolates one of the key pivot points used in the calculation.

FIG 4. shows a side view to further depict the angular dimension used in the calculation.

FIG 5. isolates the dimension that depicts the included angle of the tool tip when a ball nose cutter is used.

FIG 6. shows a full isometric 3D view of the multiple axis head and further points out another key dimension used in the calculation.

FIG 7. is an isometric view of a CNC machine tool using a multi-axes cutting tool mounted in a tool holder cutting a conceptual part.

FIG 8. is a side view of a CNC machine tool using a multi-axes cutting tool mounted in a tool holder cutting a conceptual part.

FIG 9. is a multi-axes G Code program, commonly used in CNC machining, listing the non-compensated axis positions as X,Y,Z,A,B coordinates. These coordinates are read into the system to provide a list of positions to drive the tool to and a tool number listed as "T" to provide a means to look up the user-inserted desired tool size values, as shown in FIG 1., to be used in calculating the compensation amounts.

DETAILED DESCRIPTION OF THE INVENTION

FIG 1. illustrates an operator screen where the machine operator enters the Multi-Axes Tool Compensation values. The user first starts out by entering the tool size as shown on FIG 1. in the box labeled "Size". This describes the tool diameter that will be used by the internal algorithm to compensate or adjust for this size tool when cutting a part on a multi-axes machine. On FIG 1. the box labeled "Horz" is used by the machine operator to enter the horizontal offset value that will be used to compensate for the tool location horizontally. On FIG 1. the box labeled "Vert" is used by the machine operator to enter the vertical offset value that will be used to compensate for the tool location vertically. On FIG 1. the box labeled "Height" is used by the machine operator to enter the height offset value that will be used to compensate for the tool location in regards to the tallest or length of the tool. On FIG 1. the box labeled "Wear" describes the wear amount to be deducted from the tool diameter when the tool begins to wear out so that the true tool diameter can always be compensated. On FIG 1. the boxes labeled "Custom1 and Custom2" are reserved for future user-defined values. On FIG 1. the box labeled "Corner radius" is used to enter a value by the machine operator that would be used by the internal algorithm to compensate or adjust for the radius on the end of the tool as shown in FIG 2. item (1). On FIG 1. the box labeled "Bottom angle" is used to enter a value by the machine operator that would be used by the internal algorithm to compensate or adjust for the angle on the bottom of the tool as shown in FIG 5. Dim "E" item (7). On FIG 1. the box labeled "Side angle" is used to enter a value by the machine operator that would be used by the internal algorithm to compensate or adjust for the side angle on the side of the tool. On FIG 1. the box labeled "Length" is used to enter a value by the machine operator that would be used by the internal algorithm to compensate or adjust for the actual length of the tool itself as shown in FIG 2. Dim "C" item (4). On FIG 1. the box labeled "Type" is the type or kind of tool style. The values to be entered vary from machine type to machine type and actually only provide a reference number to the machine operator as to which type of tools he uses for that particular machine type.

For instance:

Type 0=Flat Ended Mill
1=Drill Mill or Lathe
2=Diamond Insert Lathe
3=Ball Nose Mill
4=Custom
5=Circle Insert Lathe
6=Triangle Insert Lathe
7=Groove or Part Off Tool Lathe

Whereas to further describe the values to enter on FIG 1.:

Corner radius: Is the corner radius of the tool 1/2=ball nose, <= Bull nose cutter.
Bottom angle: Is the angle measured from the center tip out. Typically 28 for a drill.
Side angle: Is the angle measured on the cutter's side, like a draft angle or taper.
Length: The overall tool length.

Corner radius same as Param1
Bottom angle same as Param2
Side angle same as Param3
Length same as Param4

2=Diamond Insert Lathe
Param1 = Nose radius (Default=.03)
Param2 = Included angle of insert (Default=80)
Param3 = Mounting angle (Default=45)
Param4 = Length (Default will size to part)

5=Circle Insert Lathe
Param1 = Circle radius (Default=.03)
Param2 = Not used
Param3 = Not used
Param4 = Not used

6=Triangle Insert Lathe
Param1 = Nose radius (Default=.03)
Param2 = Mounting angle (Default=0)
Param3 = Not used
Param4 = Insert size (Default will size to part)

7=Groove or Part Off Tool Lathe
Param1 = Corner nose radius (Default=.03)
Param2 = Width (Default=.06)
Param3 = Not used
Param4 = Length (Default=2)

For a Lathe the settings mean:

SIZE: Tool nose radius
HORZ: The amount and direction of the Z offset from the tool radius center to the cutting edge.
VERT: The amount and direction of the X offset from the tool radius center to the cutting edge.
HEIGHT: Not Used
WEAR: The amount to add or subtract from the Tool Nose Radius caused by tool wear.

For non-lathe 3 or more axis single head or spindled machines the settings mean:

SIZE: Tool Diameter
HORZ: Not used
VERT: Not used
HEIGHT: Tool length offset. The amount and direction to offset the Z axis.
WEAR: The amount to add or subtract from the tool diameter caused by tool wear.

For a non-lathe 2 axis machine the settings mean (Waterjet, Laser, Plasma, EDM):

SIZE: Kerf or Tool Diameter.
HORZ: Not used
VERT: Not used
HEIGHT: Not used
WEAR: The amount to add or subtract from the tool diameter caused by tool wear.

For a non-lathe Multi-head or spindled machine.

SIZE: Tool Diameter.
HORZ: The amount or distance and direction of offset on each head in the X axis from the main head.
VERT: The amount or distance and direction of offset on each head in the Y axis from the main head.
HEIGHT: Tool length offset. The amount and direction to offset the Z axis.
WEAR: The amount to add or subtract from the tool diameter caused by tool wear.

On FIG 1. the group of boxes under the section MACHINE OFFSETS is described as follows:

Whenever you zero out the machine's home position, these are the values the machine thinks it is at. Generally, you would home the machine at zero for all axes. Ignore the values for axes you do not have.

On FIG 1. the group of boxes under the section FIXTURE OFFSETS is described as follows:

These are values for all 6 axes. Ignore the values for axes you do not have. These OFFSETs are only added when an OFFSET logic command is issued generally with G54 through G59 codes. The offset values entered for each axis will be added to the current position of the tool together with all other invoked offsets. These are typical G codes, as known by industry standards, which refer to machine offsets.

On FIG 1. the group of boxes under the section OPTIONAL SETTINGS is described as follows:

This is an array of miscellaneous settings that do not directly relate to features needed for multiple-axes tool comp. They are presented here to fully disclose how multiple-axes tool comp interacts with the various other operator settings and choices.

DRY RUN is used to switch on or off a mode in which the Z axis, Spindle and Feed mode will be disabled or not. Commonly used in test runs.

BITMAP G CODE DISPLAY will refresh the GCODE window whenever another window pops up over it.

GRAPHICS: SOLID VS WIREFRAME switches the graphics display to either a solid model or wireframe.

TOLERANCE a user-provided value in which to perform the calculations.

BLOCK SKIP CHARACTER is a user-defined character to tell the computer to skip this line of data or not.

TEACH FILE NAME is a file name in which to store all of the locations used.

SOLID STOCK Begin Z @ specifies the Z axis beginning of the solid model. Typically 0.

SOLID STOCK Extra Stock adds extra material or stock around the edges or diameter.

FANUC ARC CENTER to be calculated from Absolute, incremental or radii given arc centers.

The following paragraph is a description of the internal command set and math used internally to calculate the Multi-axes Tool Compensation algorithm. These internal commands are listed below and shown by example:

TOOLCOMP OFF	'Turns all compensation off.
TOOLCOMP LEFT	'Comps tool in 2D to the left.
TOOLCOMP RIGHT	'Comps tool in 2D to the right.
TOOLCOMP 3DCOMP	'3D comp based on vector and gouge parameter.
TOOLCOMP 3DADJUSTZ	'3D comp lifts Z axis only but keeps X,Y.
TOOLCOMP 3DOFFSET	'3D parallel offset only - based on vector and no gouge 'parameter.
TOOLCOMP 5AXIS	'5-axis comp based on vector and gouge parameter.
TOOLCOMP LLIMIT45	'Give angle which will specify a gouge to omit tool 'position.

The TOOLCOMP command enables 3D and 5-axis tool compensation and has eight possible parameters: OFF, LEFT, RIGHT, 3DCOMP, 3DADJUSTZ, 3DOFFSET, 5AXIS and LLIMIT45. The compensation value is taken from the tool parameter screen for that specific tool number.

The tool compensation is processed by the internal Multi-axes Tool Compensation algorithm when a file containing all of the locations are listed that correspond to a tool path along the original non-compensated part to cut. The positions of the original non-compensated tool path are loaded into memory.

If the tool size, height, wear, horizontal, vertical, corner radius, bottom angle, side angle, length or type is changed in the boxes depicted in FIG 1. labeled by the same names or the user edits the positions in the file or A.K.A. the G code program to reflect a change in tool comp methods, then the program will automatically reprocess new re-calculated compensated tool position.

A database is kept in the machine's memory of these positions plus ever changing conditions that it remembers by learning from what the machine can and cannot do. We regard this as an artificial intelligence algorithm element, which dynamically changes in real time.

If 3D or 5-axis tool comp is used, the file will need to include special codes on each line that will need to be compensated. The special codes are entered using the letters U,V,W or alternately A,B,C depending on the user's choice. The values following U,V,W or A,B,C represent a normalized 3D vectorized value. Vectors are common mathematical properties representing a 3D magnitude of direction pointing in 3D space. The use, knowledge and values of vectors have been used and documented for hundreds of years and are regarded as common knowledge in the mathematical field. The vector used is derived from the direction the original tool was at off the original part surface at any given point in space.

U,V,W are the end result of the compensated tool positions.
D= the distance or combined length of FIG 2. Dim "A" Item 2, Dim "B" Item 3 and Dim "C" Item 4.
Vx,Vy,Vz are the 3D vector component values.
X,Y,Z is the original non-compensated tool position

$$U = D * Vx + X$$
$$V = D * Vy + Y$$
$$W = D * Vz + Z$$

The use of the L code represents a conical angle measured from the tool tip point to the nearest obstacle from a flat 2D plane. If the user specifies an angle after LLIMIT, then the tool position move may be completely omitted by the machine if an obstacle is encountered on the part surface in order to automatically avoid gouging.

L!= the value given after the LLIMIT command.

$$L = (D / \sin(L!))$$

If $L < 0$ Then skip this move.

Else, combine this value with the D distance value to arrive at a new distance to compensate.

D=D+L

This occurs if on FIG 5. Dim "E" Item 7, which shows the included angle between the vector and the L code, is less than the value specified after LLIMIT. To turn gouge protection off, specify a zero value after LLIMIT 0.

The following mathematical algorithms are based on three values hard coded into the routine. The user does not have a box to enter these values. Each machine style and head design dynamics determine the following three values used in the compensation algorithm.

FIG 3. Item 5 depicts the key pivot point used in the algorithm. The resultant calculation affects the end position by further tilting the U,V,W in the view depicted in FIG 3. Item 5.

FIG 4. Dim "D" Item 6 depicts the key angular displacement used in the algorithm. The resultant calculation affects the end position by further tilting the U,V,W in the view depicted in FIG 4. Dim "D" Item 6.

FIG 6. Dim "R" Item 8 depicts the key rotational displacement used in the algorithm. The resultant calculation affects the end position by further rotating the U,V,W in the view depicted in FIG 4. Dim "D" Item 6.

As such:

```
Cz = Cos(Rz): Sz = Sin(Rz): Cx = Cos(Rx): Sx = Sin(Rx): Cy = Cos(Ry): Sy = Sin(Ry)

 'Z rotate, counter clockwise
 X1 = U * Cz + V * Sz: Y1 = U * -Sz + V * Cz: Z1 = W
 'Y rotate, back
 X2 = X1: Y2 = Y1 * Cx + Z1 * -Sx: Z2 = Y1 * Sx + Z1 * Cx
 'X rotate, left
 U = X2  Cy + Z2 * -Sy: V = Y2: W = X2 * Sy + Z2 * Cy
```